Fermilab

Fermi National Accelerator Laboratory
P.O. Box 500 • Batavia, Illinois • 60510

TD-99-023
April 21, 1999

Finite Element Analysis Of The Honeycomb Panels Used In Compact Muon Solenoid (CMS) Cathode Strip Chambers

S. Yadav, N. Chester and G. Apollinari
Fermi National Accelerator Laboratory, Batavia, IL, USA

A finite element analysis of a honeycomb panel has been performed to understand the deflections of the panel due to its own weight, under different support conditions. The results from the analysis have been used to provide design guidelines for adequately supporting the panels to minimize their deflection due to the gravitational force.

1. Introduction

The Compact Muon Solenoid (CMS) detector is one of the two large general purpose detectors to be installed at the Large Hadron Collider (LHC) at CERN (The European Laboratory for Particle Physics), located outside Geneva, Switzerland. Fig. 1 shows a schematic representation of the CMS detector along with the muon chambers. The endcap muon detection system (cathode strip chamber) of CMS detector is proposed to be made of seven honeycomb panels, clamped around their outer perimeter with very precise spacer bars and supported at few other points (using buttons of precise thicknesses) in between. Fig. 2 shows a drawing of one of such honeycomb panel, while Fig. 3 provides schematic representations of the cross-sectional views along the length of the panel showing the assembly of the seven panels.

The honeycomb panels being used at Fermilab for the endcap muon cathode strip chambers consist of two sheets of approximately 1/16” thick copper coated fiber glass epoxy laminate (FR4) panels, separated by and bonded to approximately ½” thick polycarbonate 1/8” (3 mm) diameter cell core by a Fermilab approved epoxy adhesive. The thickness of the polycarbonate cell core material is required not to vary more than +/-0.005” so as to meet the finished bonded panel requirements. The non-copper coated sides of the FR4 material are sanded and bonded to the cell core using a wet lay up of epoxy adhesive, completely saturating a non-woven polyester fiber sheet material used as scrim material between cell core and FR-4.

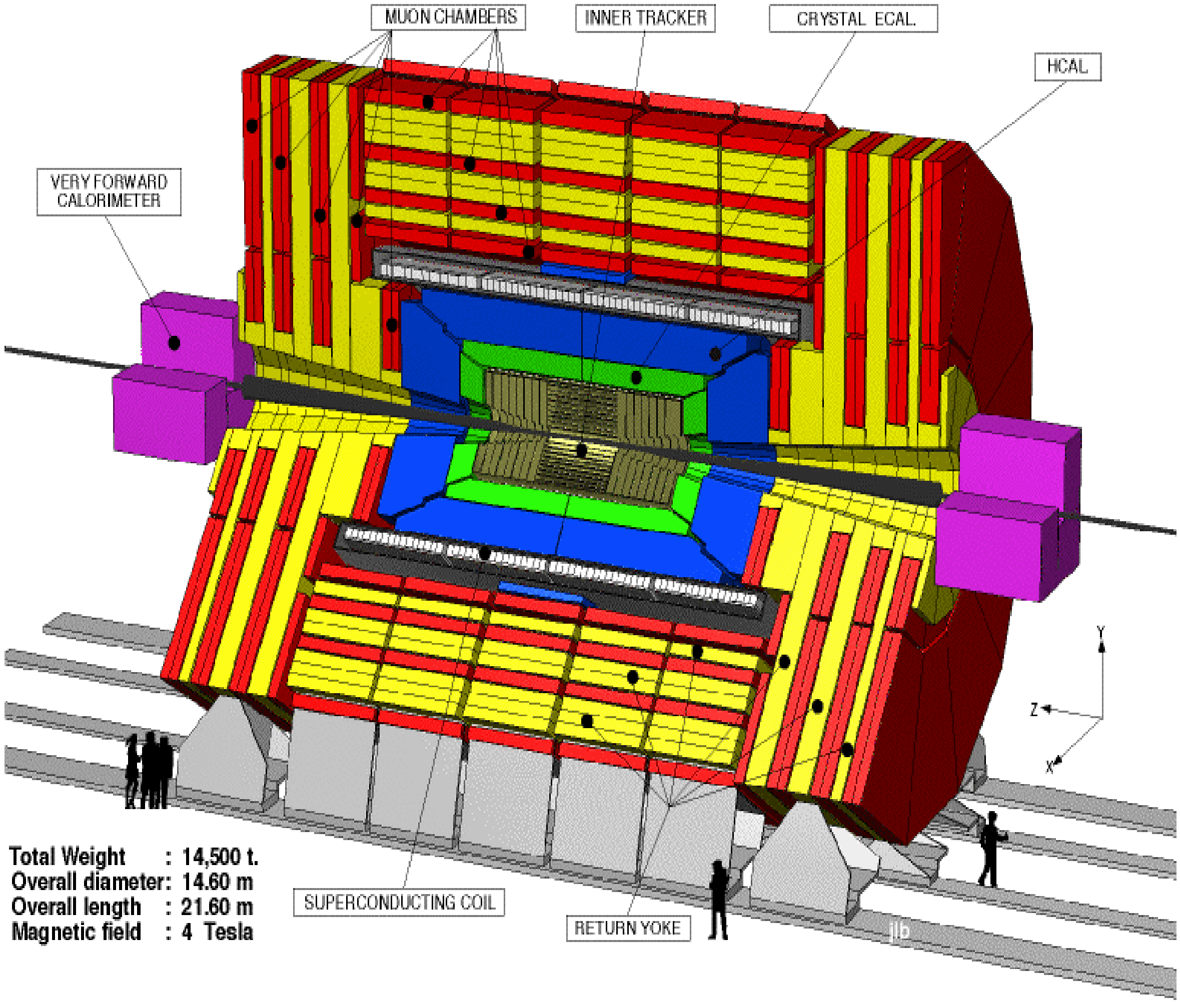

Figure 1: Schematic representation of the CMS detector showing the muon chambers.

One of the essential design constraints is the uniformity of the gap between the two panels, which should not vary by more than 0.010” (0.25 mm) over the entire surface of the panels. Also, the finished panels should be flat within 0.007” over any 12” of surface distance in any direction. However, excessive deflection (bowing) of the panels due to their own weight has been observed. This leads to a non-uniform gap between the two panels, which is not acceptable.

It is the intent of this paper to develop a finite element model, which can be used to compute panel deflections under different support conditions. Such a model can then be used to design supports for the panels such that the uniformity of the gap between two panels is within the design constraints. Theoretical calculations were performed on honeycomb panels of simple shapes (such as rectangular) to verify the finite element model used in this study. However, note that due to the trapezoidal shape of the actual panels and also due to the very tight design constraints, such simple models could not be extended to analyze the actual honeycomb panels.

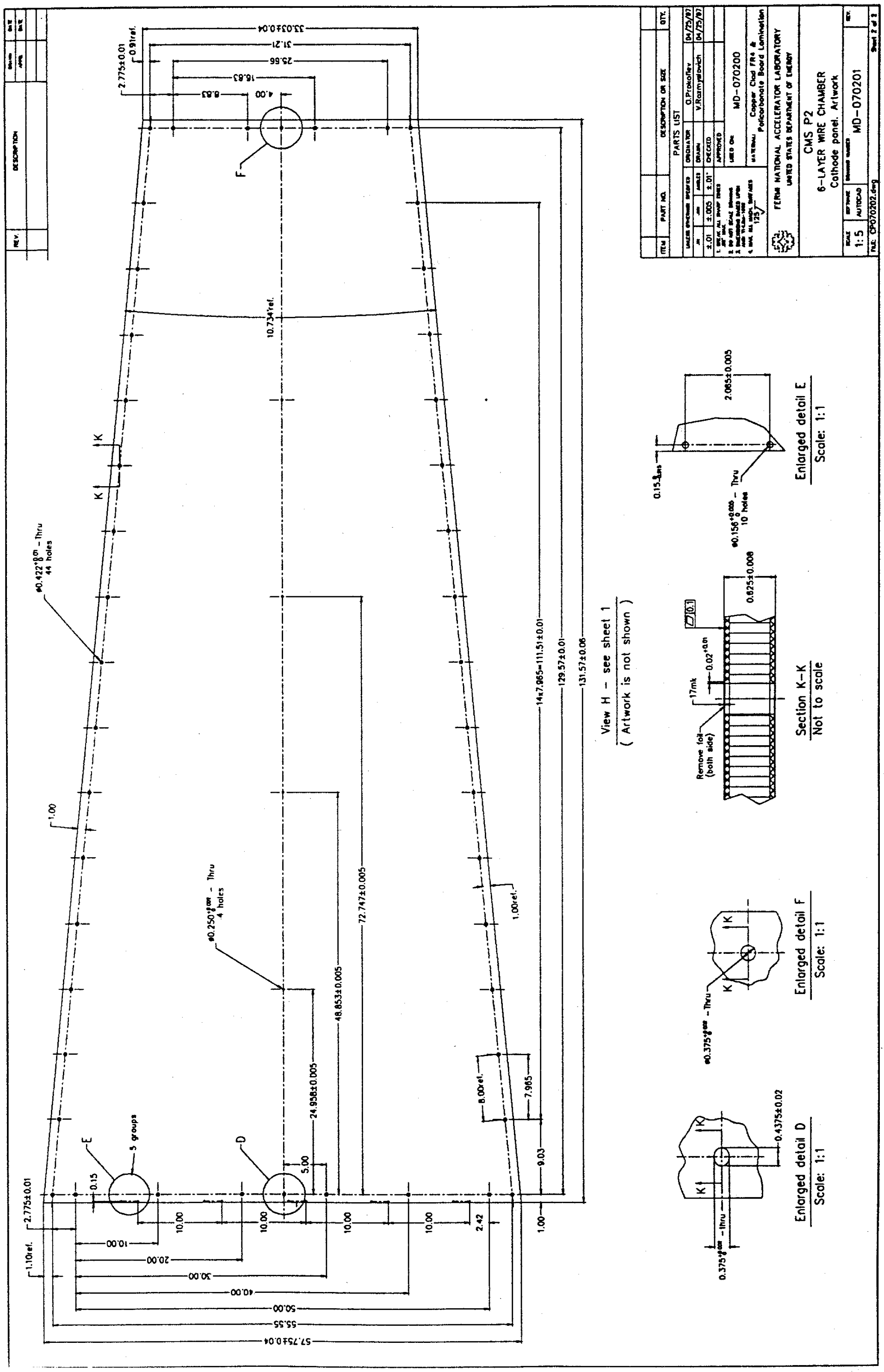

FIG. 2 : A HONEYCOMB PANEL

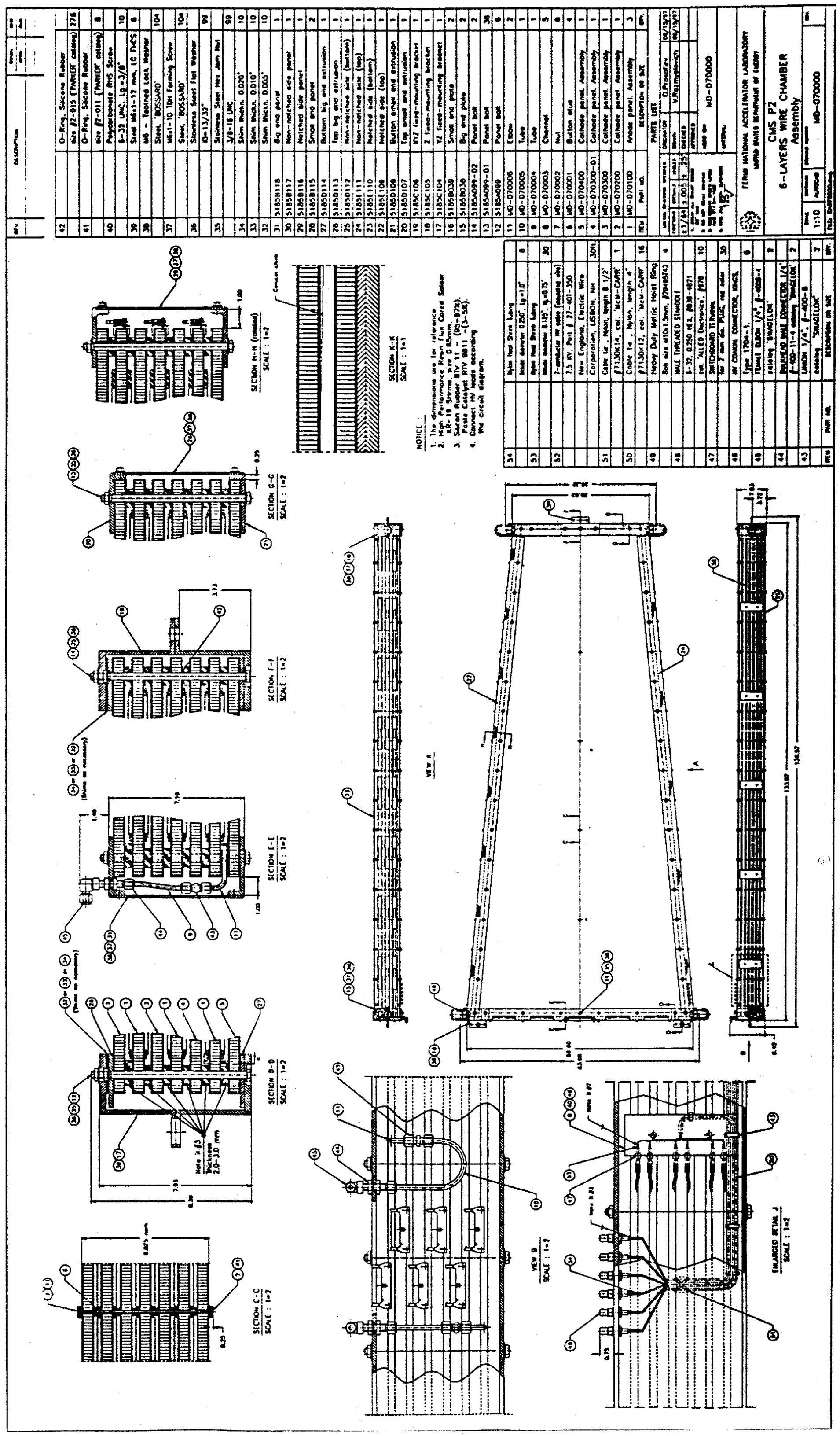

FIG. 3 : HONEYCOMB PANEL ASSEMBLY

Also, since the honeycomb material is not an isotropic material, proper care should be exercised in model development. For example, Reference [1] reports a value for the Young's modulus of the panel to be 10.1 GPa based on the experimentally measured slope of applied load vs. deflection, under a certain support condition. This number for the Young's modulus of the panel has been used by another researcher to compute the deflections of the honeycomb panel under different support conditions (including the 4-button constraint case). However, note that this is not the right approach and would certainly give erroneous results. The quantity of interest to compute panel deflections is the "*flexural rigidity*" of the panel and not some value for the panel's Young's modulus. With this in mind, the next section provides a theoretical background on bending of plates and the extension of the theory for honeycomb materials.

2. Theoretical background

2.1 Bending of thin plates

The governing equilibrium equation for the bending of thin plates is given by:

$$\nabla^4 w = \frac{P_z}{D}, \qquad (1)$$

where w is the out of plane displacement, P_z is the out of plane force acting per unit area of the plate and D is the *flexural rigidity* of the plate given by the following expression:

$$D = \frac{Et^3}{12(1-\upsilon^2)}, \qquad (2)$$

where E is the Young's modulus of the plate, t is the thickness and ν is the Poisson's ratio. The solution of the governing equation (1) is dependent on the boundary conditions of the problem and also on the applied loading. For example, for a simply supported rectangular plate of dimensions a x b subjected to a uniformly distributed load over the entire surface area of the plate, the out of plane deflection w at any point (x,y) can be obtained as:

$$w = \frac{16 P_z}{\pi^6 D} \sum_{m=1}^{\infty} \sum_{n=1}^{\infty} \frac{\sin\frac{m\pi x}{a} \sin\frac{n\pi y}{b}}{mn\left(\frac{m^2}{a^2} + \frac{n^2}{b^2}\right)^2}. \qquad (3)$$

Similarly, for a single load concentrated at any given point (ξ, η) of the plate, deflection w at any point (x, y) can be obtained as:

$$w = \frac{4 P_z}{\pi^4 a b D} \sum_{m=1}^{\infty} \sum_{n=1}^{\infty} \frac{\sin\frac{m\pi\xi}{a} \sin\frac{n\pi\eta}{b}}{\left(\frac{m^2}{a^2} + \frac{n^2}{b^2}\right)^2} \sin\frac{m\pi x}{a} \sin\frac{n\pi y}{b}. \qquad (4)$$

2.2 Bending of honeycomb panels

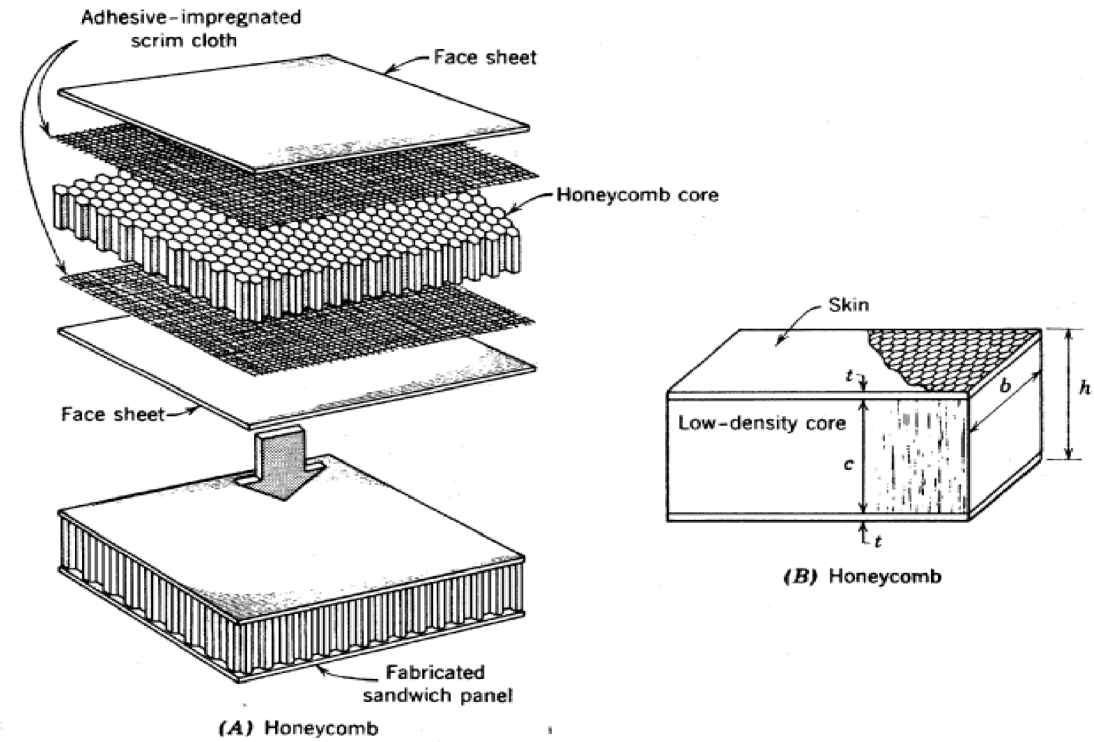

Fig. 4 Schematic of a honeycomb structure.

Fig. 4 provides a schematic representation of a honeycomb structure. Due to the nature of its construction, honeycomb panels act somewhat like I-beams: the facing correspond to flanges and carry the tensile and compressive stresses, whereas the core corresponds to the web of an I-beam and carries the shear. The core also helps to prevent buckling and wrinkling of the faces.

The formulas developed in Section 2.1 can be used for honeycomb structures too if the proper value of the *flexural rigidity D* is used. Since the core material is assumed to provide no stiffness to the structure, flexural rigidity D for the honeycomb panel can be obtained as:

$$D = \frac{E}{12(1-\upsilon^2)}\left(h^3 - c^3\right), \tag{5}$$

where E is the Young's modulus of the facing material, ν its Poisson's ratio, h is the total thickness of the honeycomb structure, and c is the thickness of the honeycomb core (as defined in Fig. 4).

3. Finite element modeling

A 2-D finite element model was set up in ANSYS to study the deflections of the honeycomb panels under their own weight and for different support conditions. The model utilizes SHELL63 elastic elements which have both bending and membrane capabilities. The element thickness was input as the total thickness of the honeycomb panel. The material properties such as Young's modulus and Poisson's ratio, for this element were taken to be the same as the material properties for the face sheets which are made of fiber glass epoxy laminates (FR4) for our case. Note that for certain non-homogeneous or sandwich shell applications, a real constant named RMI can be defined for SHELL63 elements. RMI is the ratio of the bending moment of inertia to be used to that calculated from the input thicknesses. For our case RMI can be defined as:

$$RMI = \frac{D_{honeycomb}}{D_{solid}} = \frac{\frac{E}{12(1-\upsilon^2)}\left(h^3 - c^3\right)}{\frac{E\,h^3}{12(1-\upsilon^2)}}. \tag{6}$$

A value of RMI computed from equation (6) was used to compute out of plane deflections of the honeycomb panel under their own weight for different support conditions.

As a first step, computations were performed on a simply supported rectangular beam with a uniformly distributed normal load applied to it. Theoretical solutions for this case can be obtained fairly easily. The finite element solution obtained for this simple case was verified with the theoretical computations and the two results matched very well. This gave us confidence in our model so that we could extend it for the case of actual panels and study their deformation behavior under different support conditions.

Since a simply supported panel would deform more than a panel that is held rigidly at its ends, a case of simply supported panel was investigated. The weight of the panel was 60 lbs (27.3 kg) which was assumed to be uniformly distributed over the entire surface area (of 3.84 sq. meter) of the panel. This gave a force per unit area acting on the panel of 70 N/m^2. Fig. 5 shows the contour plot of out of plane displacement for this case. A maximum out of plane displacement of 0.437 x 10^{-3} m (0.017") is obtained for this simple case.

Referring back to Fig. 2, we can see that the panel is also supported at 4 other points in between using buttons of precise thicknesses. The area of the panel supported at these buttons was originally designed to be 0.1 inch square for each button support. However, on assembling the seven panels, it was observed that there was excessive bowing of the panels near these button supports due to the very high normal stresses acting on the panel at these four points of support. Thus, it was of interest to know as to what forces act on these 4 points of support in the actual assembly of the honeycomb panels. Also, to get an estimate of the localized panel deformation due to these forces, some experiments were performed in the Material Development Laboratory, where small samples (such as 1” x 1”) of the honeycomb material were loaded using the original buttons and a load-deflection curve was obtained.

A finite element model of a single honeycomb panel was set up, where the panel was simply supported at the 4 locations corresponding to the location of the buttons, in addition to being simply supported all along its perimeter. The deflection of the panel due to its own weight was again studied. Fig. 6 shows the contour plot of out of plane displacement for this case. Also, the reaction forces acting on the four buttons were obtained to be 39 N, 33.4 N, 26.5 N and 30.3 N. From the solution for a single panel, an estimate was made for the reaction forces at the bottom most panel of an assembly of seven honeycomb panels. Based on this estimate of maximum reaction force and experimental data obtained at the Material Development Laboratory, it was concluded that the present design of the buttons was not sufficient to adequately support the panels. It was decided to increase the area of support provided by the buttons (so as to reduce the compressive stresses) based on the finite element results and the experimental data. A set of panel was assembled based on this modified design of the buttons and was observed to perform within the design constraints.

References

[1] C. Bradford, E. Dijak et al., “CSC panels: flatness and Young’s modulus measurements,” CMS-TN-95/094, 1995.

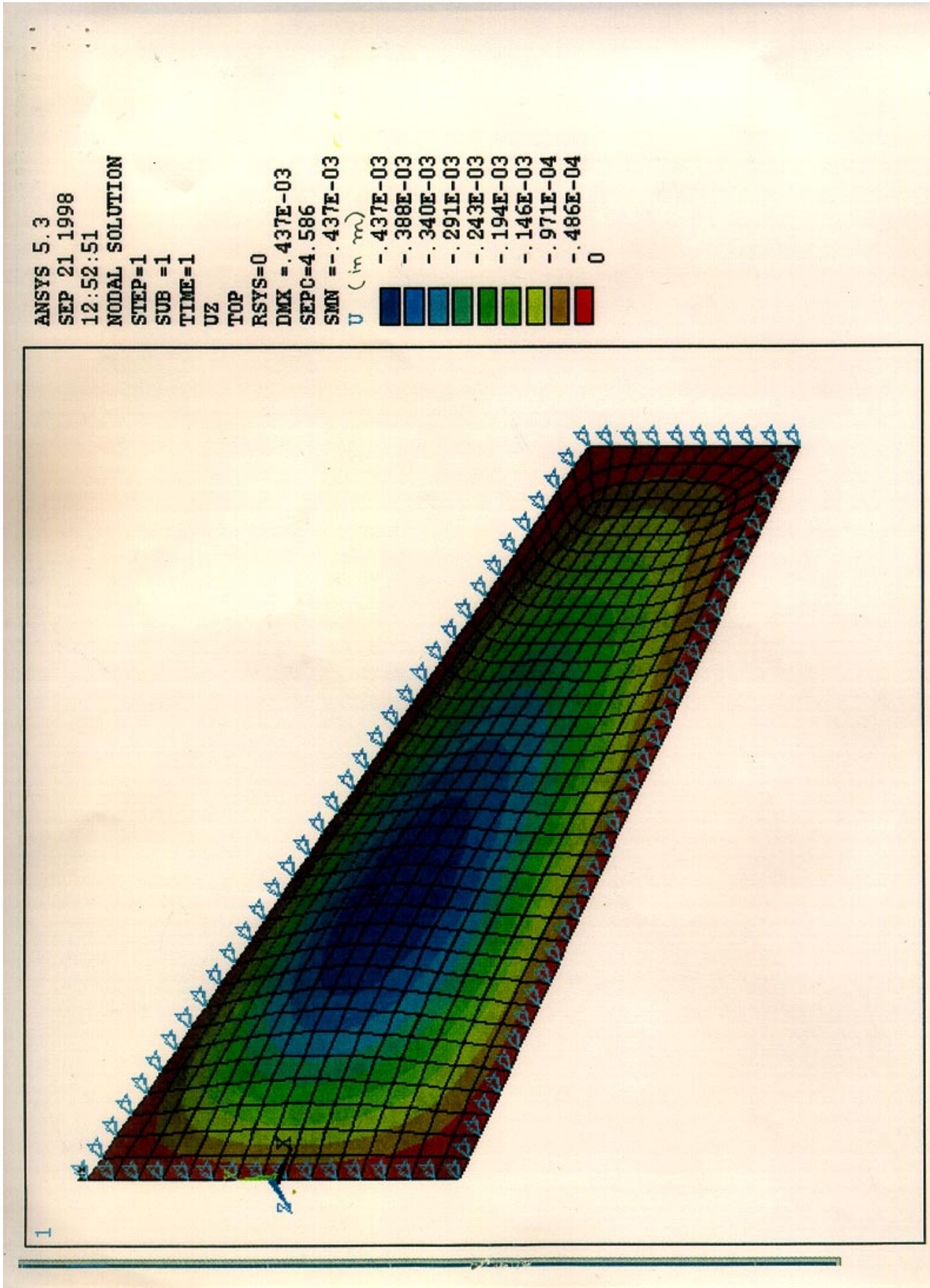

FIG.5: CONTOUR PLOT OF OUT OF PLANE DISPLACEMENT (SIMPLY SUPPORTED PANEL)

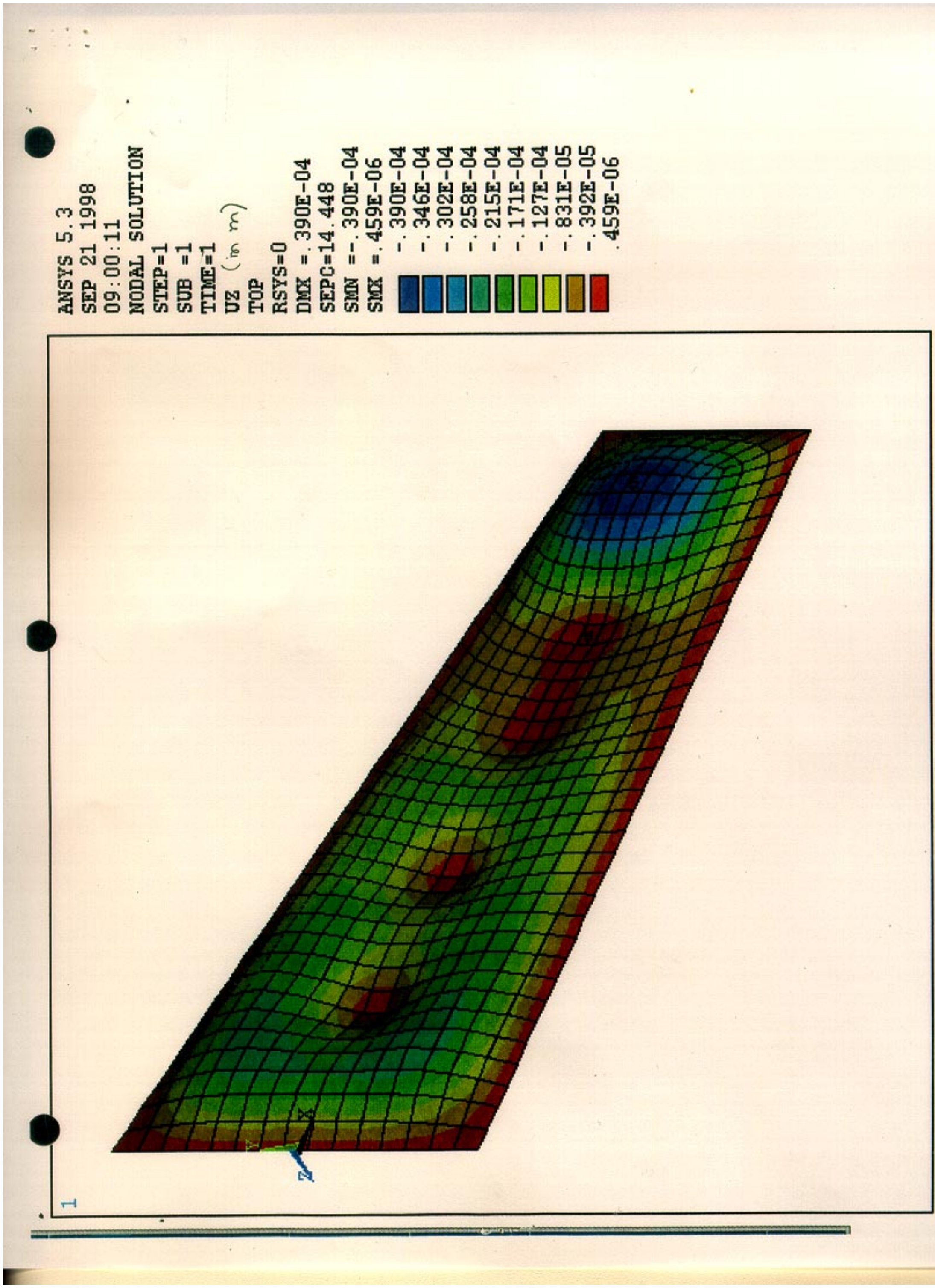

FIG. 6 : CONTOUR PLOT OF OUT OF PLANE DISPLACEMENT (SIMPLY SUPPORTED + 4-BUTTON SUPPORT)